Proceedings of the 44th National Conference on Fluid Mechanics and Fluid Power
December 14-16, 2017, Amrita University, Amritapuri Campus, Kollam, Kerala, India

FMFP2017–PAPER NO. 175

Issues in Simulating Central Plume in Bottom Gas Injection using OpenFOAM

Veeraraghavan Kannan
SASTRA University, Thanjavur
613401
Email: veeraraghavank96@gmail.com

P.R. Naren
SASTRA University, Thanjavur
613401
Email: prnaren@scbt.sastra.edu

Abhishek Dutta
KU Leuven, Leuven
B-3000
Email: abhishek.dutta@kuleuven.be

Abstract

Injection of inert gas into molten metal (liquid) is a significant step in steel manufacturing process to remove slag from molten metal, to enhance mixing and the rate of chemical reaction. The hydrodynamics of this process can be compared with a bubble column, ubiquitous in most (bio)chemical process industries. In this study, air-water two phase model is simulated to understand the hydrodynamics of bottom gas injection into quiescent liquid . The effect of different drag models in predicting the central plume is observed. The equations are solved numerically using *twoPhaseEulerFoam* solver present in open-source CFD software OpenFOAM-v4.0. The efficacy of the drag models in capturing the flow is determined by comparing the results with the established results of Ma et al. [1] and Davidson [2].

Keywords: *CFD, two-phase flow, central plume, drag models.*

1.0 INTRODUCTION

Multiphase flows can be seen in most of the industrial processes such as chemical, petrochemical, biochemical and metallurgical processes. The last few decades witnessed rapid development of numerical methods based on the continuum (Euler/Euler) and the discrete (Euler/Lagrange) approaches to simulate unsteady and dispersed (here gas–liquid) flows in order to better understand the physics and improve the process performance.

The plume formed when a gas is introduced into a liquid can play an important role in acclimatizing chemical and thermal reactions of the mixture (Bernard et.al. [3]). This is the basic principle used in steel manufacturing units such as ladle where inert gas is bubbled through porous plug into molten metal to enhance alloying or metallic treatment practices.

Plume generation is a salient feature in gas injection into liquid flows. Drag force exerted on the bubble(s) is principally important in simulating this feature. This work deals with studying the effect of different drag models on predicting the central plume with RANS based turbulence model. The centreline velocity, gas holdup at different section of the two different flow domains (Fig. 1 and Fig. 2) is compared to arrive at some understanding. The main objective of this work is to study the efficacy of RANS based model to predict the central plume in a bottom gas injection process using open source CFD software. Though LES has been widely used in recent times for such flows, it is not adopted for the present work as it is computationally more expensive than RANS.

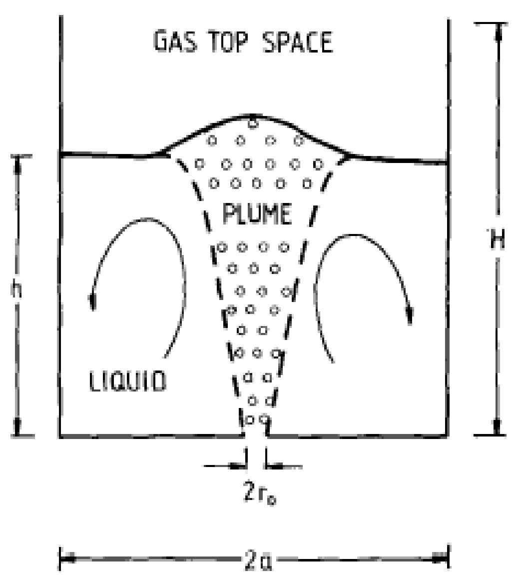


Fig 1 2D Computational Domain for Davidson [2] showing schematic sketch of central plume Dimensions H = 60 cm, h = 40 cm (water level), $2r_o$ = 0.635 cm, 2a = 50 cm.

Table 1 Summary of the state-of-the-art in bottom gas injection into liquid

S. No.	**Reference**	**Category (E/ S / ES)**	**CFD Model**	**Primary Phase**	**Secondary Phase**	**Geometry (Length × Breadth × Height)**	**Remarks**
1	Rabha & Buwa [16]	S	3D	Water	Air	150 mm × 30 mm × 15 mm	The rise behavior of single/multiple bubbles in liquids of different properties imposed with linear shear
3	Neto.et.al [15]	E	-	Water	Air	1.2 m × 0.8 m	Effect of number of nozzles on bubble characteristics
4	Buwa &Ranade [14]	ES	3D	Water	Air	200 mm × 50 mm × 1200 mm	Plume oscillation frequency, bubble passage frequencies are studied
5	Buwa & Ranade [13]	ES	3D	Water	Air	200 mm × 50 mm × 900 mm	Effect of superficial gas velocity and sparger configurations
6	Freire.et.al [12]	E	-	NaCl	Air	1 m × 1 m × 1 m	Dependence of deflection angle of plume with modified Weber and Froude number
7	Sato [11]	ES	3D	Water	Air	435 mm × 435 mm × 300 mm	Relationship between intrusion depth and stratification intensity, gas flow rate, bubble size
8	Bernard.et.al [3]	S	3D	Water	Air	Height = 3.05 m Dia. = 30.5 m	Validated the bubble slip velocity with available experimental data
9	Pan.et.al [10]	ES	2D	S: Water E:Steel	S: Nitrogen E:Argon	500 mm × 300 mm × 2 mm	Effect of single bubble shape and bubbles interaction.
10	Mazumdar & Guthrie [9]	ES	2D	S: Water E:Steel	S: NA E:Argon	S: Height = 0.93 m Dia. = 1.12 m Nozzle Dia. = 6.35 mm E: Height = 3.04 m Dia. = 3.65 m Nozzle Dia. = 20.28 mm	Effect of side walls and surface baffles on rising plume

E: Experiment S: Simulation NA: Not Available

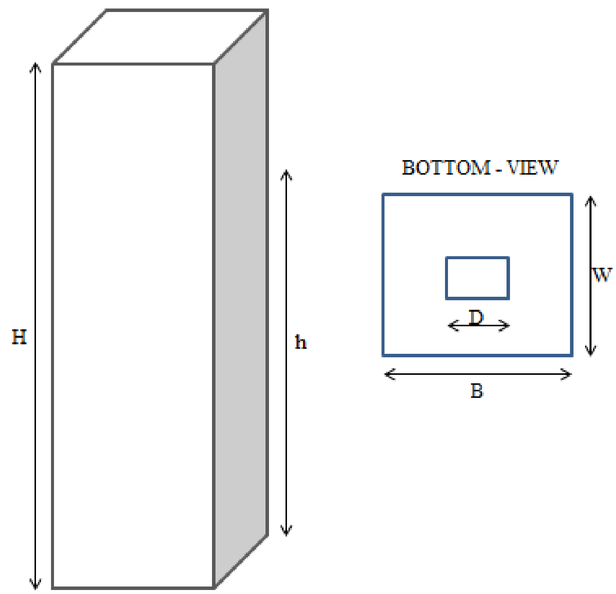


Fig 2 3D Computational Domain for Ma et al. [1]. Dimensions: H=50 cm, h=45cm, B=W=15cm, D=3.7cm

1.1 Previous Work

Previous experimental and computational studies on bottom gas injection in liquid are shown in Table 1. The effect of drag model in simulating the central plume was not addressed in most of these reported works.

2.0 METHODOLOGY

2.1 Governing Equations

The governing equations of continuity and momentum with the appropriate Reynolds stress closure were solved in order to simulate the flow. Euler-Euler approach was used to carry out the simulation. Turbulence was modelled using the mixture k-ε turbulence model available in OpenFOAM. The interfacial forces considered in the present work were lift, drag, virtual mass and turbulent dispersion force. Except the drag force, rest of the interphase exchange force were kept same across all simulated cases. The lift force coefficient was given by Tomiyama [4]. The virtual mass force coefficient was set as 0.5. The governing equations are not produced here for sake of brevity but can be obtained from Drew [5]

2.2 Simulation Details

In the present work, simulations were carried out for two domains given in Fig. 1 and Fig. 2. Unstructured hexahedral mesh was used to mesh both the computational domain. The computational domain were discretised with uniform cells with $\Delta x = \Delta y = 4$ mm resulting in 48000 cells for Fig. 1 and $\Delta x = \Delta y = \Delta z = 5$ mm resulting in 81000 cells for Fig. 2.

No slip condition was used for both gas and liquid phase at solid boundaries. The normal velocity of each phase and the normal gradient of the void fraction were also set to zero at boundaries. At the top boundary of the computation domain velocity of liquid was assumed to be zero to prevent liquid from escaping the boundary and the gradient of velocity for air is set to zero.

2.3 Solution Procedure

The governing equations were solved under transient condition using open source CFD code OpenFOAM (v 4.0) with *twophaseEulerFoam* solver. All the equations were solved in segregated manner with the SIMPLE algorithm for pressure-velocity term. A second-order scheme was used for discretisation. Gradients of pressure and velocity fields were discretised with second order Gaussian finite volume integration using linear interpolation. Residuals of continuity were monitored to ascertain numerical convergence. In all the simulations, a physical variable (velocity of air in centreline) was also monitored for convergence. It is noteworthy to mention that time step was kept very low of the order of 10^{-5} to maintain proper CFL number range, and to ensure convergence of the solution.

3.0 RESULTS AND DICUSSIONS

The objective of the work is to compare the effect of various drag models in predicting the flow pattern with gas injected into liquid using RANS based turbulence model. Void fraction and the centreline vertical velocities are compared with Ma et al. [1] and Davison [2]. The drag models considered for simulation in this paper are namely Schiller-Neumann model [6], Tomiyama-Correlated model [7] and Tomiyama-Analytic model [8] which are by default available in OpenFOAM.

3.1 Effect of Drag with Ma et al. [1] simulation

3.1.1 Mid-Plane Velocity Profile

Fig. 3 shows the contour of instantaneous velocity of air for the three different drag models at t=40s. The velocity profile along the horizontal at the mid-plane from bottom is plotted in Fig. 4. Symmetric profile was observed by Ma et.al. [1]. Present simulations also showed a nearly symmetric profile for all drag models tested in the present study. However, the peak value is over estimated by the simulation and also the location of peak was slightly offset from that of results reported by Ma et. al [1].

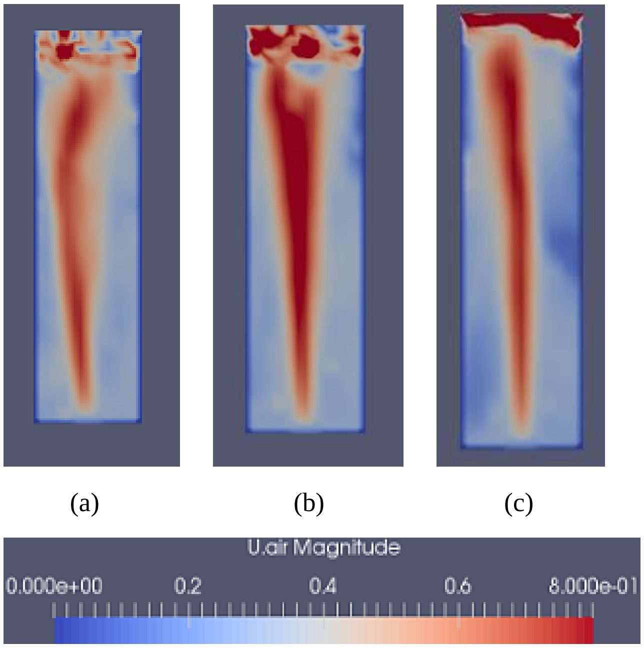


Fig 3 Instantaneous Air Velocity Contour at the mid-Plane for three different drag models at t=40s : (a) Tomiyama-Analytic (b) Tomiyama-Correlated (c) Schiller-Neumann

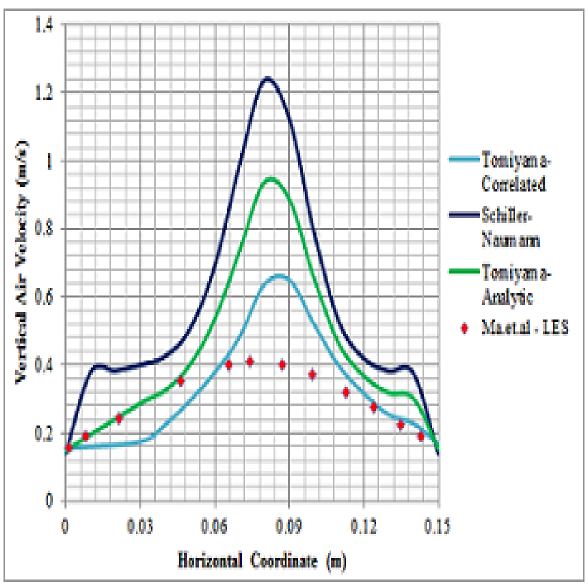


Fig 4 Velocity profile of Air at mid-plane along horizontal

3.1.2 Mid-Plane Void Fraction Profile

The contour of instantaneous void fraction for the three different drag models is shown in Fig. 5. From Fig. 6 it can be inferred that there is not much difference in the flow pattern on varying the drag model in simulation. Void fraction of air along the horizontal plane at mid-plane is shown in Fig. 6. The void fraction shows similar symmetry behaviour along the horizontal, the value increases and attains peak, then it starts decreasing. A sharp peak is observed in simulations done in OpenFOAM as compared to a more wide spread profile observed by Ma et.al. The peak values are overestimated and the location of maxima is offset in reference to the reported values by Ma et al. [1].

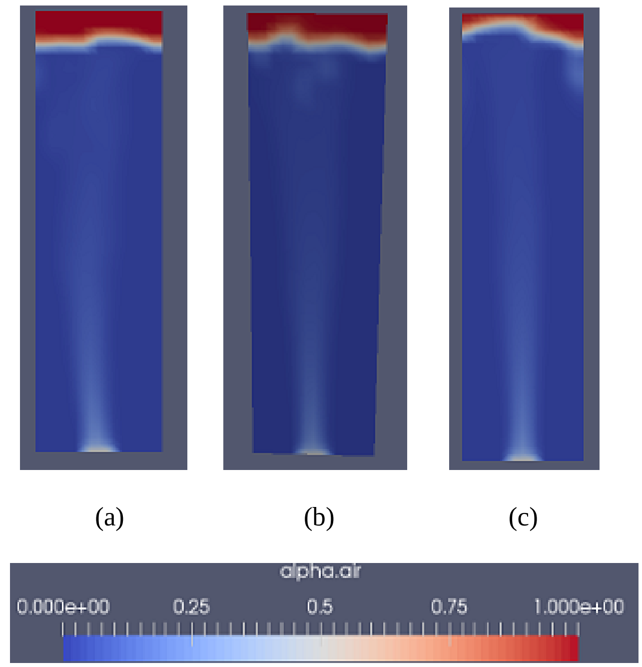


Fig 5 Instantaneous Void Fraction of Air at the mid-Plane for three different drag models at t=40s : (a) Tomiyama-Analytic (b) Tomiyama-Correlated (c) Schiller-Neumann

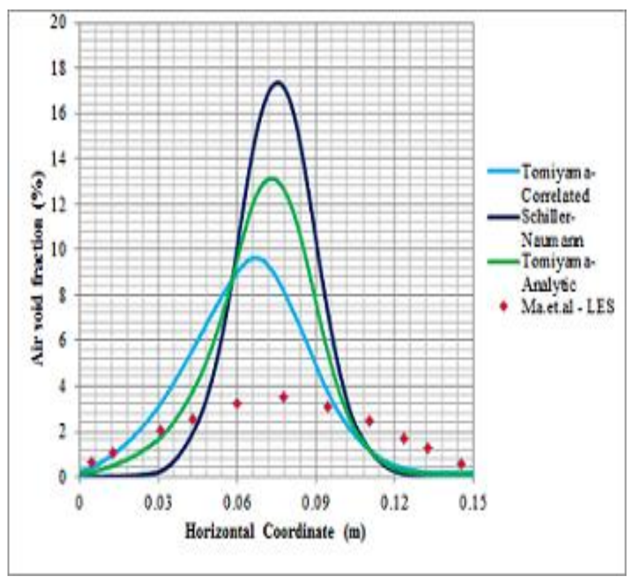
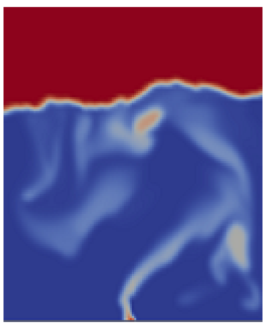


Fig 6 Air void fraction at mid-plane along horizontal

3.2 Effect of Drag with Davidson [2] simulation

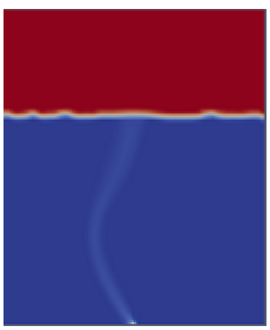
3.2.1. Centreline Void Fraction

Fig. 7 shows the instantaneous void fraction of air for different drag models. It can be seen from Fig. 8 that the centreline void fraction decreases from top to bottom and similar pattern were reported by Davidson [2]. It can also be seen from Fig. 8 that simulated results showed large deviations in comparison to the published results of Davidson[2]. Deviation was found to be larger for Schiller-Naumann drag model with a maximum deviation of about 60% at approximately z =14 cm from bottom of the domain. Tomiyama-Correlated drag model gave a deviation of about 35.7% compared to other drag models considered in this study.



(a)

(b)



(c)

Fig 7 Instantaneous Void Fraction of Air at the mid-Plane for three different drag models at t=40s : (a) Tomiyama-Analytic (b) Tomiyama-Correlated (c) Schiller-Neumann

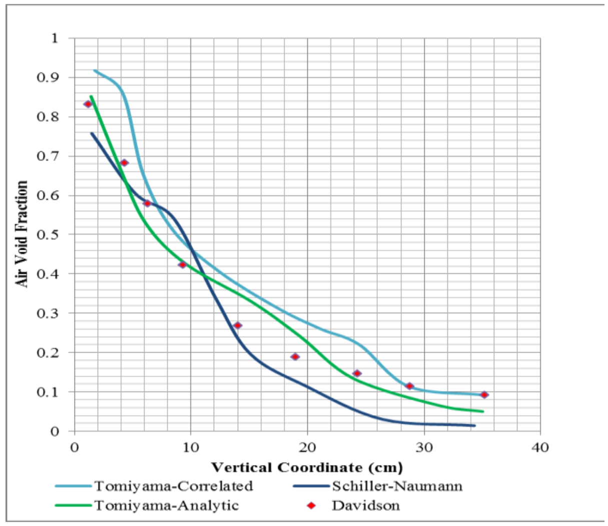


Fig 8 Centreline void fraction profile of air

3.2.2 Centreline Vertical Velocity

The instantaneous contour of air velocity for different drag models is shown in Fig. 9. Fig. 10 shows the comparison of the vertical velocity along the centreline of the computational domain between different drag models and the result reported by Davidson [2]. It can be seen clearly from the plot that the simulated results have a large deviation. The maximum deviation of 60% is observed for Schiller-Neumann drag model at approximately z = 20 cm and Tomiyama-Correlated drag model gives deviation of about 32% among the different drag model considered in the simulation.

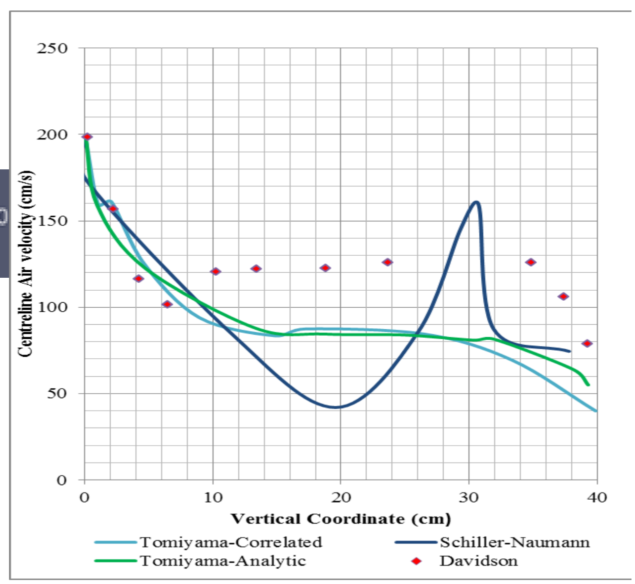


Fig 10 Centreline vertical velocity profile of air

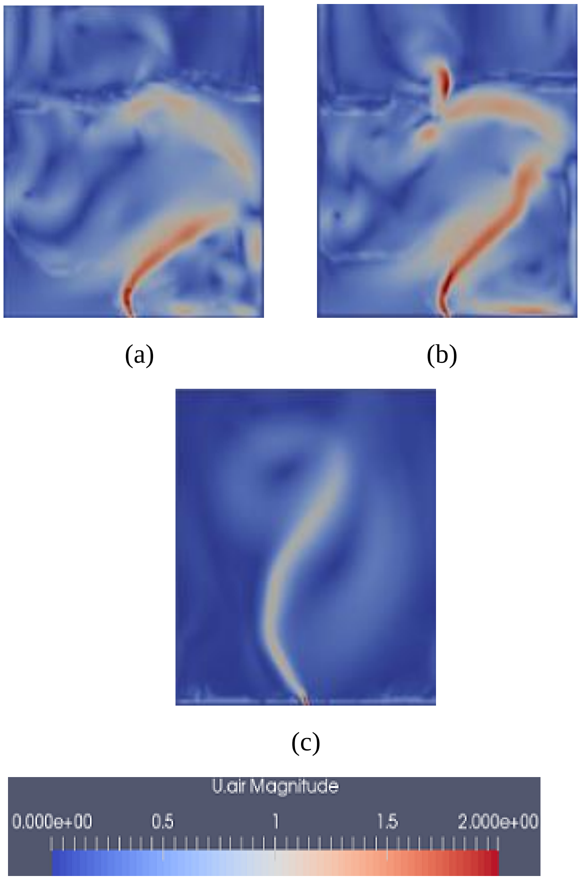


Fig 9 Instantaneous Air Velocity Contour at the mid-Plane for three different drag models at t=40s : (a) Tomiyama-Analytic (b) Tomiyama-Correlated (c) Schiller-Neumann

4.0 CONCLUSIONS

In the present work, Finite Volume Method (FVM) based approach is used to simulate hydrodynamics of air injected into water using open source code OpenFOAM-v4.0. Drag models namely Tomiyama correlated, Tomiyama Analytic, and Schiller Neumann were used to simulate the model and the results are compared with two case studies from the literature. Simulated results show that selection of drag model plays a crucial role in simulating the hydrodynamics in case of bottom gas injection. In the present case the outlet boundary is modelled by specifying zero velocity for the liquid phase. Alternate approach could be use of degassing boundary condition for the outlet. Further work also involves assessment of turbulence models in simulating bubble plume and assessment of numerical schemes in simulating two phase flow in OpenFOAM.

REFERENCES

1. Ma T, Lucas D, Ziegenhein T, Frohlich.J.,Deen.N.G,(2015) Scale-adaptive simulation of a square cross sectional bubble column, Chemical Engineering Science,131,101-108.
2. Davidson M. (1990). Numerical calculations of two-phase flow in a liquid bath with bottom gas injection: The central plume. Applied Mathematical Modelling, 14, 67 - 76
3. Bernard RS, Maier RS, Falvey HT. (2000). A simple computational model for bubble plumes. Applied Mathematical Modelling, 24, 215 – 233
4. Tomiyama A., Tamai H., Zun I., Hosokawa S., (2002), Transverse migration of single bubble in simple shear flows, Chemical Engineering Science , 57(11) , 1849-1858.
5. Drew, D. A. (1983), Mathematical modelling of two-phase flow. Ann.Rev. Fluid Mech. 15, 261-291
6. Schiller, L. and Naumann, Z., A drag coefficient correlation. *Zeitschrift des Vereines Deutscher Ingenieure*, 77-318 (1935)
7. Tomiyama, G.P. Celata, S. Hosokawa and S. Yoshida, (2002)"Terminal velocity of single bubbles in surface tension dominant regime", *Int. J. Multiphase Flow* **28**, pp. 1497-1519.
8. Tomiyama T., Katoka I., Zun I., Sakaguchi T., (1998), Drag coefficients of single bubbles under normal and micro gravity conditions. ISME Int. Journal series B Fluids and Thermal Engineering, 41, 472-479.
9. Mazumdar D, Guthrie RIL. (1985). Hydrodynamic Modelling of some gas Injection procedures in Ladle Metallurgy Operations. *Metallurgical Transactions B*, 16B, 83 – 90
10. Pan SM, Chiang JD, Hwang WS. (1998). Simulation of Large Bubble/Molten Steel Interaction for Gas-Injected Ladle. *Journal of Materials Engineering and Performance*, 8, 236 – 244
11. Sato K, Sato T. (2001). A study on plume behaviour in stratified water. *Journal of Marine Science and Technology*, 6, 59 – 69
12. Freire APS, Miranda DDE, Luz LMS, Franca GFM. (2002). Bubble plumes and Coanda effect. *International Journal of Multiphase Flow*, 28, 1293 – 1310
13. Buwa VV, Ranade VV. (2002). Dynamics of gas-liquid flow in a rectangular bubble column: experiments and single/multi-group CFD simulations. *Chemical Engineering Science*, 57, 4715 – 4736
14. Buwa VV, Ranade VV. (2004). Characterisation of Dynamics of Gas-Liquid flows in rectangular bubble columns. *American Institute of Chemical Engineers*, 50, 2394 – 2407
15. Neto IEL, Zhu DZ, Rajarathnam N. (2008). Air injection in water with different nozzle. Journal of Environmental Engineering, 134, 283 – 294
16. Rabha SS, Buwa VV. (2010). Volume-of-fluid (VOF) simulation of rise of single/multiple bubbles in sheared liquids. *Chemical Engineering Science*, 65, 527 – 537